ELSEVIER
CrossMark
Available online at www.sciencedirect.com
ScienceDirect
Energy Procedia 85 (2016) 390 – 398
Energy Procedia

Sustainable Solutions for Energy and Environment, EENVIRO - YRC 2015, 18-20 November 2015, Bucharest, Romania

Auto radiators - Study regarding air flow along the channels

Angela Pleşa[a]*, Oana Giurgiu[a], Daniel Banyai[a]

[a]Technical University of Cluj Napoca, Muncii Bulevard, no.103-105, Postcod: 400641, Cluj-Napoca, România

Abstract

In this paper will be presented the results of the measurements and experimental data processing performed on aluminium auto radiators manufactured in Romania. Also, will be exposed numerical simulations of air flow through an auto radiator, from the models studied. It will be study the convective heat exchange efficiency through Nusselt number and intensity of heat transfer by forced convection of air through the fins with sinusoidal profile using Colburn criterion. Further, the numerical simulation of air flow through a mini channel created by a fin with sinusoidal profile, considered on the width of the device, highlights the heat diffusivity through Prandtl number, along this channel, depending on the particularities of its geometric shape.

Peer-review under responsibility of the organizing committee EENVIRO 2015
Keywords: Micro channels with sinusoidal fins; Nusselt; Colburn and Prandtl numbers; heat distribution through air stream.

1. Introduction

This paper is part of a larger study of aluminum auto radiators manufactured in Romania [1] and follows on research conducted on these devices by Prof. Mihai Nagi from the Polytechnic University of Timisoara together with PhD. Eng. Paul Ilieş and Eng. Vlad Martian from factory of aluminum radiators RAAL Bistrita Romania [2,4].

The objectives of this paper are:

- establishing relations criteria for calculating the Nusselt and Colburn number, specific for sinusoidal channels of air flow with fixed height h_n = 8.8 mm, variable pitch p_n = 4 mm, respectively p_n = 6.5 mm and different lengths of these channels L_{air} = 30, 45 65, 95 and 115 mm;

* Corresponding author.
E-mail address: angela.plesa@termo.utcluj.ro.

Peer-review under responsibility of the organizing committee EENVIRO 2015
doi:10.1016/j.egypro.2015.12.266

- analysis of convective heat exchange by Nusselt number and intensity of heat transfer by Colburn number;
- numerical analysis of air flow through a mini channel, that highlights:
  - boundary layer separation and
  - heat diffusivity along the channel.

In section 2 of this paper are exposed models of auto radiators for which measurements were performed. After processing measured data were plotted graphs of variation of Nusselt numbers in function of Reynolds criteria.

The authors have developed a calculation program using Engineering Equation Solver software. With this program were generated criteria for calculating equations for Nusselt and Colburn criteria (equations 1, 2 and 4) exposed in section 3 of this paper. Also, have been drawn graphs of variation of these criteria in function of Reynolds for fins step number 4 mm 6.5 mm respectively.

In section 4 is presented the CFD simulation of air flow through a channel pn = 4 mm and length Lair = 30 mm. CFD simulation results were compared with those obtained by calculation program developed in software EES.

Nomenclature

α	convection factor
C_S	Smagorinsky constant
d	the distance to the nearest wall
$d_{2\ ech}$	hydraulic diameter for air flow through the cross section of the channel
f	Colburn factor
h_n	height of air channels
k	Kármán constant
L_{air}	lengths of the air channels
μ_t	the turbulent viscosity
Nu	Nusselt number
p_n	fins step
Pr	Prandtl number
Re_{2c}	Reynolds number of the air for a channel
St	Stanton-criteria
$w_{2\ c}$	air velocity for one channel

2. Measurements and results interpretation

Auto radiators are parts of the cooling systems used in the compound of the air compressors, the energy transportation systems, machine tools, hydraulic devices, etc. The heat exchangers that were studied by the authors were water-air type.

The experimental stand was design in order to give the possibility of measuring the parameters (pressure, flow, and temperature) needed for predicting the thermal global coefficient and also the pressure drop. Another reason was that the same stand to be used for measuring the parameters for different heat exchangers.

The experimental stand is presented in fig.1. It has three main circuits: air, hot water and power. The measured parameters for each heat exchanger are: mass flow $\dot{m}_i$ [kg/s] , pressure [MPa], agent's „i" the temperature at the entrance and at the exit in the heat exchanger*:* $t_i^{'}$, $t_i^{''}$ *[^{0}C].*

The length of the air channel is different for each heat exchanger. The other geometrical configurations were kept constant. Five heat exchangers with different length of the air channel *(L_{air}= 30, 45, 65, 95,* respectively 115 mm) for a variable pitch p_n *= 4 mm* and *6.5 mm* were tested: *L30-4 (L_{air} = 30 mm, p_n = 4 mm), L30-6.5, L45-4, L45-6.5, L65-4, L65-6.5, L95-4, L95-6.5, L115-4, L115-6.5.*

Fig.1 The experimental stand [1]

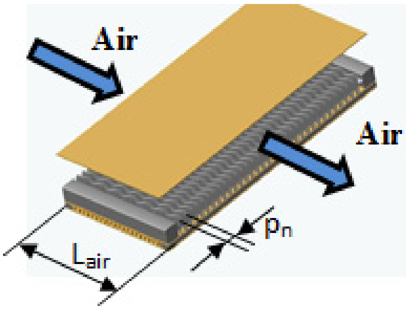

Fig. 2 Strip fins [1]

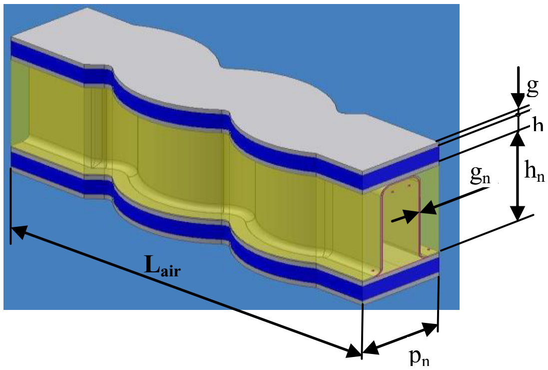

Fig. 3 Dimension of air channel [1]

Knowing the air convection coefficient allows us the calculation of convective heat exchange efficiency factor Nusselt through its relationship definition. From the graphic representation (Figure 4), it is noted that by the linearly approximately increase of Nu with increasing Reynolds number for all types of heat exchanger studied, with values in the range (11...44), due to the convective heat flux dependence expressed by (α_2).

For values of the Reynolds number in the range of laminar flow is remarkable influence of resistance at thermal transfer transmitted on selected characteristic length *(d_{2ech})*, which is why Nu_2 values are very close to all exchangers no matter the step fins. For $Re > 1000$ the influence of this resistance becomes dominant. For a given length L_{aer}, Nusselt invariant suffer a influence of the convective flux, having a similar behaviour when increases with Reynolds. For *Re = constant* and different lengths *for* L_{aer} Nusselt criteria decreases with increasing the length of air channel, no matter of size of p_n step (Figure 4a and b).

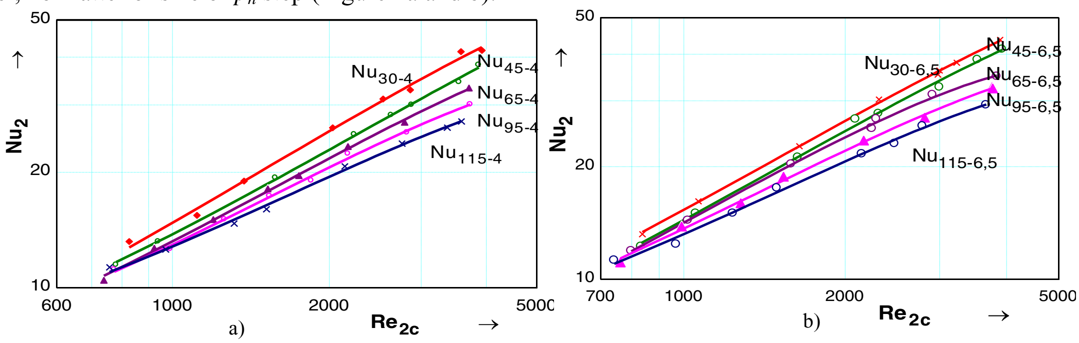

Fig.4 Nusselt variation in function of Reynolds for different steps: a) $p_n = 4$ mm, b) $p_n = 6.5$ mm [1]

3. Mathematical analysis

Variation of Nusselt criterion based on the Reynolds number determined the author of this paper to find criteria relationships for calculating this criterion, expressing both dependency Nu_2 number of criteria Re_2 and characteristic patterns of geometric dimensions studied [2]. Based on research conducted, the author determined the following criteria for setting relations that allows convective calculating of flow through sinusoidal channels [3]:

$$\text{for } p_n = 4\ mm, \quad Nu_{2c} = C \cdot \mathrm{Re}_{2c}^{x} \cdot \mathrm{Pr}^{1/3} \cdot \left(\frac{d_{2ech}}{L_{d-aer}} \right)^{y} \cdot \left(\frac{p_n - g_n}{h_{cr}} \right)^{-0.32} \tag{1}$$

$$\text{for } p_n = 6{,}5\ mm, \quad Nu_{2c} = C \cdot \mathrm{Re}_{2c}^{x} \cdot \mathrm{Pr}^{1/3} \cdot \left(\frac{d_{2ech}}{L_{d-aer}} \right)^{y} \cdot \left(\frac{p_n - g_n}{h_{cr}} \right)^{-0.66} \tag{2}$$

where the constant C, x and y are values derived according to the type of exchanger, with relative errors less than 3.5% compared to the calculated values. $h_{cr} = h_n/2$ represents the critical fin height, lengths to which the heat flux propagates through the material of the fin. Graphs of the variation of the corrected functions in equations (1) and (2) are drawn in Figure 5 for a step 4 mm and in figure 6 the step of 6.5 mm.

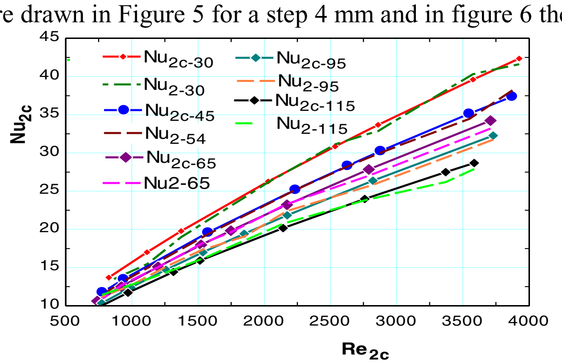

Fig.5 – Nusselt criteria corrected for p_n = 4 mm [1]

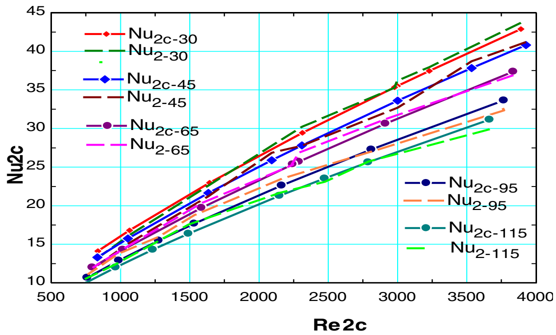

Fig.6 – Nusselt criteria corrected for p_n = 6,5 mm [1]

To compare the performance of extended surfaces, literature use Colburn factor that shows the intensity of heat transfer measure by forced convection [3, 5].

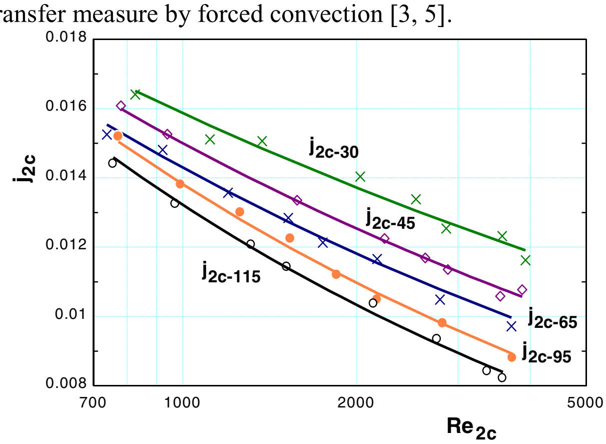

Fig.7 –Colburn function corrected for p_n= 4 mm[1]

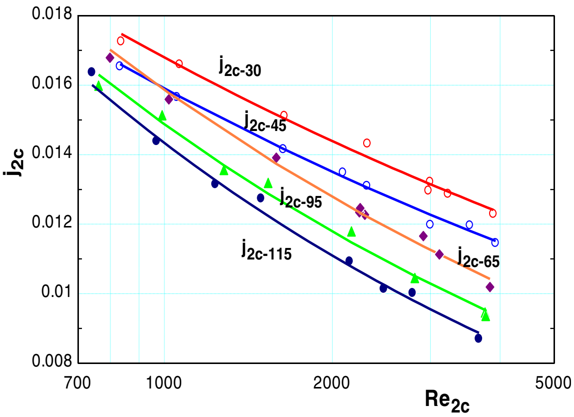

Fig.8 –Colburn function corrected for p_n=6,5 mm[1]

In the graphs shown in Figures 7 and 8 is represented the variation by points based on Reynolds of the Colburn criterion resulted from calculation, for a 4 mm step (Figure 7) and 6.5 mm (Figure 8), and through continuous lines appropriate corrective functions. Whatever the measure of adopted step, it is observed inverse dependence of Colburn function for air against proper Reynolds number for a flow through a channel. Thanks to the small diameters equivalent flow, but mostly because of very low kinematic viscosity, air velocity leads to lower growth ratio.

Stanton-criteria function, defined by relation (3) [6], depends directly proportional by this report and inversely proportional to the number Prandtl, which recorded slight variations in the range (0.76 ... 0.8). This leads to decrease in the range Stanton (0.02 ... 0.01) with increasing speed. This decrease is recorded also for the number ($Pr_2^{2/3}$) in the same terms of speed variation.

$$St = \frac{Nu}{\mathrm{Re} \cdot \mathrm{Pr}} \tag{3}$$

In terms of math, function Colburn vary directly with the product $\left(St \cdot \mathrm{Pr}^{2/3}\right)$ and having both the product terms recording a decrease to an increase Reynolds means that the Colburn criterion will have a decreasing trend in the studied flow. Most heat exchangers with large areas are designed and adapted based on correlations obtained experimentally and that can be expressed graphically or as simple criteria relations. For exchangers studied in this paper, the author develops a correlation as a criterion of the dependence relations by Reynolds number, as follows:

$$j_{2c} = a \cdot \mathrm{Re}_{2c}^{b} \tag{4}$$

where the correlation coefficients a and b have values depending on of exchanger type (p_n respectively L_{aer}).

Tabel 1 - Constant of criteria relation (4) for p_n = 4 mm [3]

	L30-4	L45-4	L65-4	L95-4	L115-4
a	0,067522	0,0895681	0,0955	0,137	0,154411
b	-0,2097	-0,2587	-0,275	-0,425	-0,355826
Er_j	< 2,7 %	< 2,2 %	< 2,8 %	< 2,3 %	< 2,9 %

The influence of traverse step from corrugated fins over the values of the heat transfer factor j variation is evident in the chart $j_2 = f(Re_{2c})$, through the superiority of corresponding values for heat exchangers with a 6.5 mm step, compared to the ones with 4mm step. This is due to a more intense convective heat transfer for exchangers with 6.5mm step. Relationship criterion (4) facilitates comparison of thermal performance for various models of fins that are found in compact heat exchanger construction.

Mathematical modelling results obtained from the general calculation of the program conceived by the authors, highlight the following aspects of convective heat transfer through exchangers with compact plates and fins:

- Convection phenomenon is reduced as the channel length increases, so, for the fins type *L30_4*, $\alpha_2 = (82 \div 259)$ W/m²K, and the fins type *L115_4* records $\alpha_2 = (73 \div 172)$ W/m²K; for L_{aer} big, the difference between convection coefficient corresponding to p_n = 6.5mm and p_n = 4mm is greater than the small values of L_{aer}, and for $Re < 1000$ numbers the step size has no influence;
- Effectiveness factor of convective heat exchange *(Nu_2)* has values very close for all exchangers regardless of the fins step; there is a higher power for heat transmission for fins with 6.5 mm step than those with increments of 4 mm step.

4. Numerical analysis

The authors used FLUENT software in order to simulate the flow through the channels of a heat exchanger. Large Eddy Simulation (LES) method was used for the CFD study on the geometry of plate and fins aluminium heat exchanger channel, presented in figure 1. The same geometry was tested on an experimental setup developed by the authors. The experimental configuration gives the authors the possibility to measure the parameters involved in the performances of the heat exchangers, such as: pressure, pressure drop, mass flow, temperature and humidity. As it is known the flow through the channels of heat exchangers have to be turbulent. It is possible to analyse the entire turbulent field using Direct Numerical Simulation method (DNS). This method is inefficient for engineering problems due to the fact that considering the difference between maximum and minimum values of the Reynolds number, $\mathrm{Re}_t^{3/4}$, the cells dimensions have to be proportional with $\mathrm{Re}_t^{9/4}$. DNS method simulates the unsteady flow, with small time steps. These two aspects are more than DNS can support. There is another method, based on Reynolds Averaged Navier-Stokes, equations, known as RANS, in which the equations of motion for fluid are basically time-averaged. The RANS equations are primarily used to describe turbulent flows and are widely used in research of fluid flow and heat transfer [7-8]. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate time-averaged solutions to the Navier–Stokes equations.

4.1. Mathematical model

Large Eddy Simulation method allows solving the large vortices and modelling the small vortices. The main characteristics of *LES* method are:

- the momentum, mass and energy are scalars and they are carried out by large vortices,
- large vortices are dependent by the geometry and frontier conditions,
- small vortices are not depending by the geometry and tend to be isotropic,

Solving only large vortices the regarding for meshing is less restrictive for LES method than for DNS. The time step is proportional with the turn over time of the vortices, which is also an unrestrictive condition.

The equations characteristic for LES method are obtained either by filtering Navier-Stokes equations, by erasing the low energy vortices (Eq. 5) [9]:

$$\overline{\Phi}(x) = \int_D \Phi(x') \cdot G(x, x') dx' \tag{5}$$

where G is the filtering function that gives the values of the vortices [10].

$$G(x, x') = \begin{cases} \dfrac{1}{V}, & x' \in v \\ 0, & x' \notin v \end{cases} \tag{6}$$

The cell volume V, generates the filtering [10].

$$\overline{\Phi}(x) = \frac{1}{V} \int \Phi(x') dx', \quad x' \in v \tag{7}$$

After filtering Navier-Stokes equations are [10]:

$$\frac{\partial p}{\partial t} + \frac{\partial}{\partial x_i}(\rho \overline{u}_i) = 0 \tag{8}$$

and

$$\frac{\partial}{\partial t}(\rho \overline{u}_i) + \frac{\partial}{\partial x_j}(\rho \overline{u}_i \overline{u}_j) = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{u}_i}{\partial x_j}\right) - \frac{\partial \overline{p}}{\partial x_i} - \frac{\partial T_{ij}}{\partial x_j} \tag{9}$$

where

$$T_{ij} = \rho\overline{u_i u_j} - \rho\overline{u}_i\overline{u}_j \tag{10}$$

$$T_{ij} - \frac{1}{3}T_{kk}\delta_{ij} = -2\mu_t\overline{S}_{ij} \tag{11}$$

where μ_t is the turbulent viscozity and S_{ij} is [10]:

$$\overline{S}_{ij} \equiv \frac{1}{2}\left(\frac{\partial\overline{u}_i}{\partial x_j} + \frac{\partial\overline{u}_j}{\partial x_i}\right) \tag{12}$$

Smagorinsky-Lilly model which was defined by Smagorinsky and improved by Lilly, gave the equation [10]:

$$\mu_t = \rho L_S^2\left|\overline{S}\right| \tag{13}$$

where L_s is the medium length of the cells, expressed by [10]:

$$L_S = \min\left(kd \cdot C_S V^{1/3}\right) \tag{14}$$

$$And\ \left|\overline{S}\right| \equiv \sqrt{2\overline{S}_{ij}\,\overline{S}_{ij}} \tag{15}$$

where:
C_S is Smagorinsky constant, k – Kármán constant,
d – the distance to the nearest wall

4.2. CFD results

The numerical simulation applied to the geometry of the channel presented in figure 2 was made using FLUENT software. The results, presented in figure 9, show the variation of the air temperature in the channel. In Table 2 are presented the air temperature values at the exit of the channel obtained by measuring ($T_{e\,2}$ and by numerical simulation ($T_{e\,2}^*$) for different values of velocity of air flow through the channel ($w_{2\,c}$). Note that the values are very close, with a relative error below 0.55% [1].

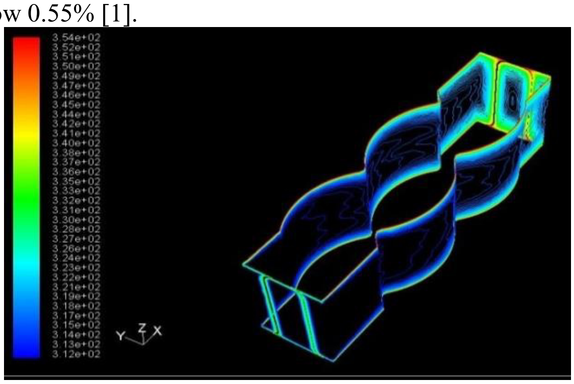

Fig.9 Variation of the air temperature in the channel [1]

Air velocity resulting from measures	Air temperature at the exit of the channel		Error
	measure	CFD	
w_{2c}, [m/s]	T_{e2}, [^{0}C]	T_{e2}^*, [^{0}C]	E_{Te2}, [%]
17.6	53.37	53.51	0.0413
16.16	54.21	54.47	-0.1041
13	56.10	56.35	0.1518
11.71	57.95	57.6	-0.5460
9.4677	60.30	60.6	-0.5122
6.476	62.88	62.47	-0.1229
4.973	56.99	57.25	0.3810
3.745	60.35	60.72	0.2698

These temperatures are used to determine air thermo physics characteristics which will be considered in calculations of invariants Reynolds, Prandtl, Nusselt and Colburn. Once with air heating during crossing the channel are improved the thermo physics performance and it's certified through calculated and simulated results of Prandtl number indicated in Figure 10.

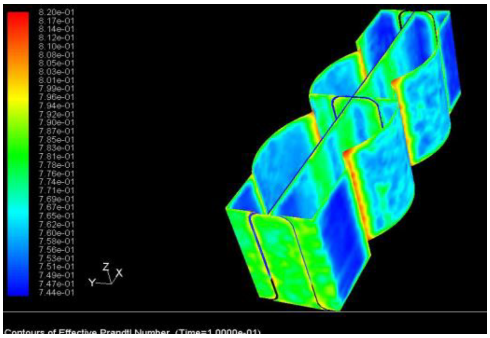

Fig.10 – Variation of the Prandtl number [1]

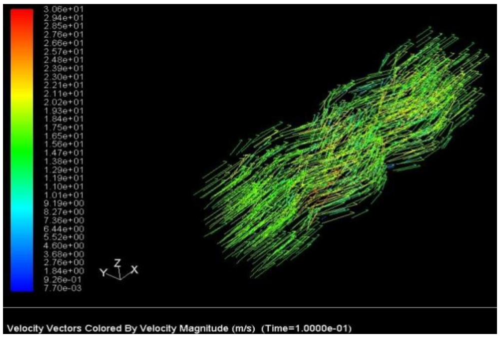

Fig.11- Velocity vectors distribution[1]

The velocity vector of the air stream flowing through the channel breaks down on three directions of flow (xyz) equivalent ($L_{air} - p_n - h_n$) (Figure 11). Note that the maximum amplitude of the sinusoidal channel ($y = p_n / 2$), the main air flow direction of vectors that have maximum velocity marked in red in Figure 11, which leads to dislocation of the boundary [3].

This phenomenon occurs both on the left wall, and on the right fin profile. Incidence of both current jets leads to the formation of vortex in fluid mass. These swirl areas are visible in Figure 12 and highlight convection, default Nusselt and Colburn numbers that will have maximum values in these areas.

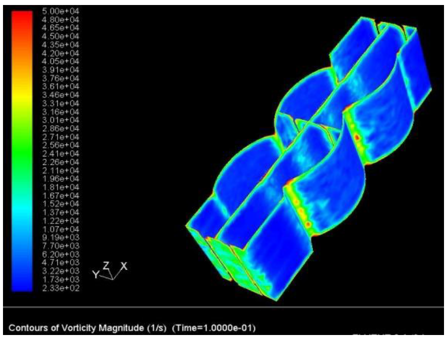

Fig.12 - Nusselt number and vortices distribution [1]

The intensity of the velocity vector, highlight transformation of kinetic energy of incident current into pressure energy which then dissipates along the channel.

5. Conclusion

This paper presents 10 types of auto radiators produced in Romania on which were carried out measurements that have been processed and introduced into a mathematical modelling program. This allowed the authors the development of criteria equations for calculating Colburn and Nusselt numbers that take into account the geometrical characteristics of this type of fins. These equations have a calculation error below 3% compared with those deduced from measurements.

Numerical simulation highlights the different movement of air layers through the channel, and also boundary dislocation and incidence jets created. All this leads to vortex formation in fluid mass, a phenomenon that intensifies the heat exchange, especially in the curvature areas of air channel. Numerical simulation results have a 0.6% deviation compared to the results obtained with the mathematical model developed, which certifies its validation.

Acknowledgment

This work was supported by Grant of the Romanian National Authority for Scientific Research, CNCS, UEFISCDI, PN-II-PT-PCCA-2011-3.2-1212 - Advanced strategies for high performance indoor Environmental Quality in Operating Rooms.

References

[1].Angela Plesa, Studiul curgerii aerului prin radiatoarele auto din aluminiu. Ed. U.T.Press, Cluj-Napoca, 2014, ISBN 973-606-737-004-1, 2014.
[2].M. Nagi, P.Ilies, and V. Martian, An experimental approch for air flow inside the heat exchanger's wavy channe. Conf.nat. termo, part intern, Ploiesti, Romania, 2007.1: p. 140-143.
[3].Azis, A, Mc. Fadehen, (2006) "Some new solution for extended surface heat transfer using symbolic algebra", Heat Transfer Engineering Vol.26, No. 8, oct.2006, p. 15-23.
[4].M. Nagi, I.D. Negru, Cercetări privind perfectionarea sub aspect functional al schimbatoarelor de caldura compacte din aluminiu, cu nervuri onduate, destinate motoarelor cu ardere interna, Conferinta Nationala de Termotehnica (II), Timisoara, Romania. 1994, p. 74.
[5].Rush, T.A., Newell, T.A., Jacobi, A.M., (1999)"An Experimental study of flow and heat transfer in sinusoidal wavy passages", Int. J. Heat Mass Transfer 42, p. 1541-1553.
[6].T. Carpiuc, T. Madarasan, C. Silasi, Stabilirea invariantilor de similitudine pentru proiectarea recuperatoarelor de caldura prin radiatie, Buletinul Stiintific al Institutului Politehnic Cluj Napoca, Seria Chimie-Metalurgie (21), 1978, p. 56-61
[7].Meslem, A., et al., Optimization of a Lobed Perforated Panel Diffuser - A Numerical Study of Orifice Arrangement. The International Journal of Ventilation, 2012. 11(3): p. 255-270.
[8].Meslem, A., et al., Comparison of turbulence models in simulating jet flow from a cross-shaped orifice. European Journal of Mechanics - B/Fluids, Volume 44, Pages 100–120, 2014.
[9].www.fluent.com – Fluent Users Guide